

# Introduction to Finite Element

ENGR 45901/55901/75901  
Spring Semester 2024

Dr. A. Aziz

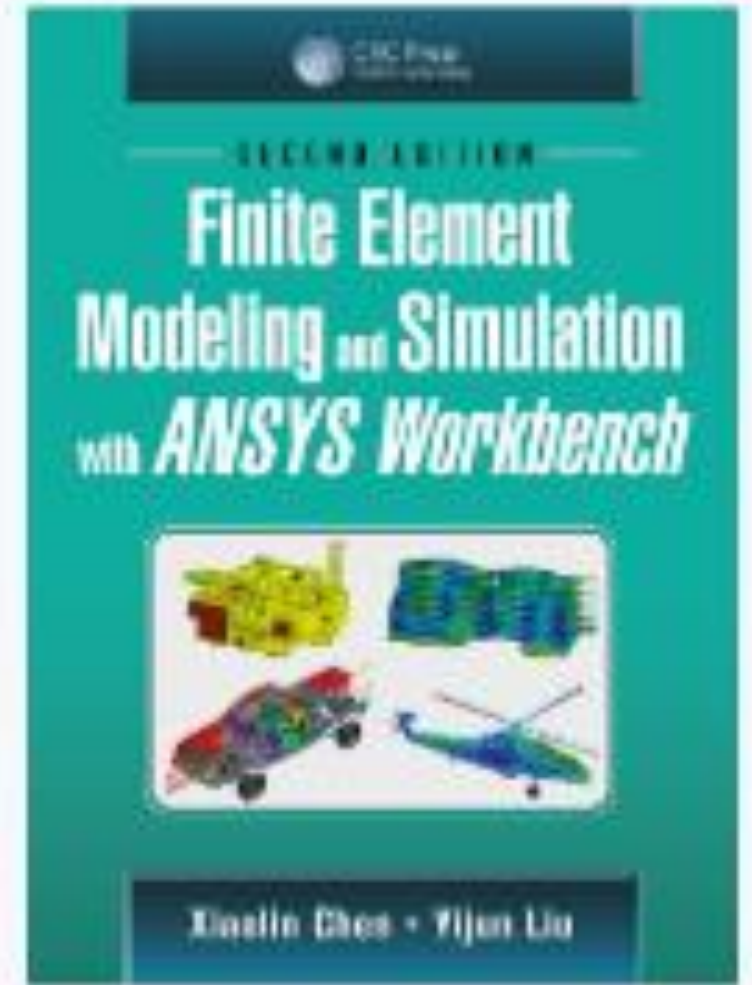
College of Engineering and Aeronautics

# Introduction

## Chapter 1

Finite Element Modeling  
and Simulation with  
ANSYS Workbench,  
Second Edition, 2nd...

Xiaolin Chen; Yijun Liu



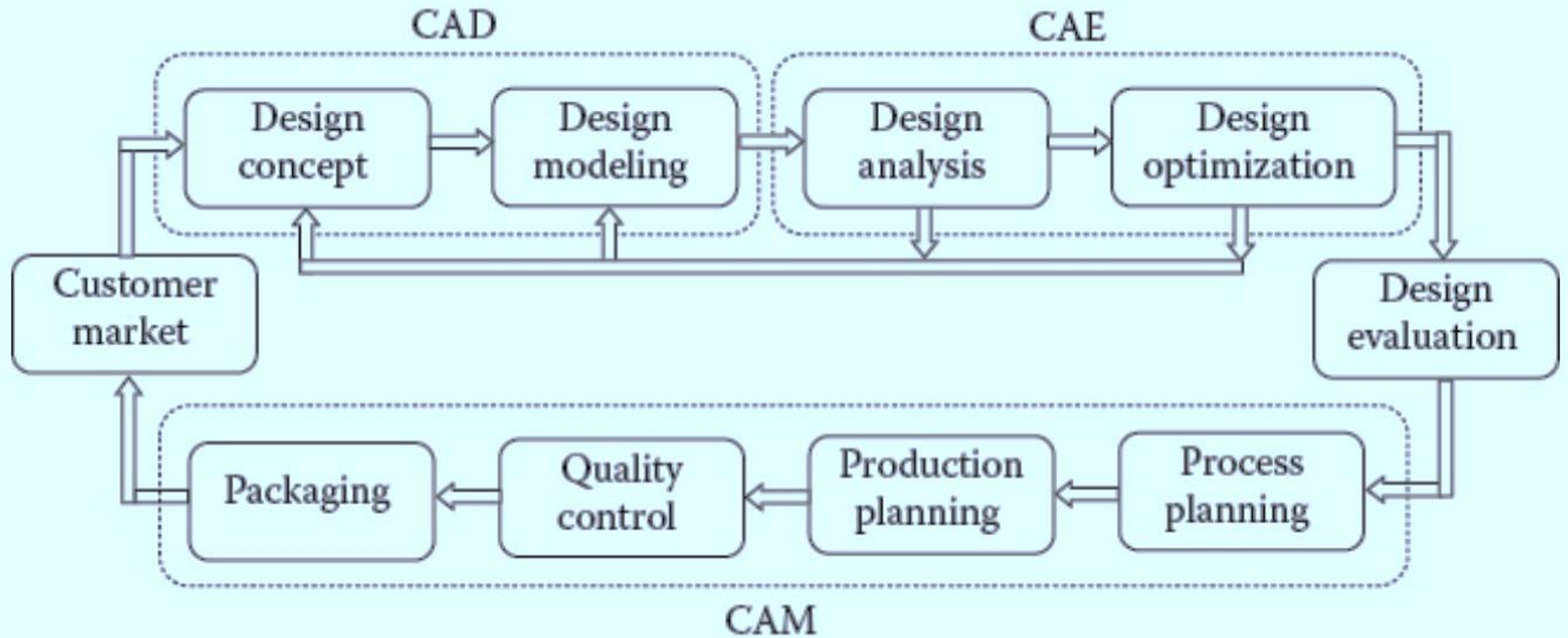
## 1.1.2 Finite Element Applications in Engineering

- ❑ The FEM can be applied in solving the mathematical models of many engineering problems,
- ❑ from stress analysis of truss and frame structures or complicated machines, to dynamic responses of automobiles, trains, or airplanes under different mechanical, thermal, or electromagnetic loading.
- ❑ There are numerous finite element applications in industries, ranging from automotive, aerospace, defense, consumer products, and industrial equipment to energy, transportation and construction, as shown by some examples in Table 1.1.
- ❑ The applications of the FEA have also been extended to materials science, biomedical engineering, geophysics, and many other emerging fields in recent years.

## 1.1.1 Why FEA?

Computers have revolutionized the practice of engineering.

- Design of a product that used to be done by tedious hand drawings has been replaced by computer-aided design (CAD) using computer graphics.
- Analysis of a design used to be done by hand calculations and many of the testing have been replaced by computer simulations using computer-aided engineering (CAE) software.
- Together, CAD, CAE, and computer-aided manufacturing (CAM) have dramatically changed the landscape of engineering.
- For example, a car, that used to take five to six years from design to product, can now be produced starting from the concept design to the manufacturing within a year using the CAD/CAE/CAM technologies.



**FIGURE 1.2**

A sketch of the computer-aided product development process.

- Among all the computational tools for CAE, the FEM is the most widely applied method or one of the most powerful modern “calculators” available for engineering students and professionals.
- FEA provides a way of virtually testing a product design. It helps users understand their designs and implement appropriate design changes early in the product development process.
- The adoption of FEA in the design cycle is driven by market pressure since it brings many benefits that will help companies make better



## 1.1.2 Finite Element Applications in Engineering

The FEM can be applied in solving the mathematical models of many engineering problems,

- ☐ From stress analysis of truss and frame structures
- ☐ Or complicated machines,
- ☐ To dynamic responses of automobiles,
- ☐ To, or airplanes under different mechanical, thermal, or electromagnetic loading.

## 1.1.2 Finite Element Applications in Engineering

There are numerous finite element applications in industries ranging from automotive,

- ☐ Aerospace,
- ☐ Defense,
- ☐ Consumer Products,
- ☐ And industrial equipment to energy, transportation and construction,
- ☐ As shown by some examples in Table 1.1.



## 1.1.2 Finite Element Applications in Engineering

**TABLE 1.1**

Examples of Engineering Applications Using FEA

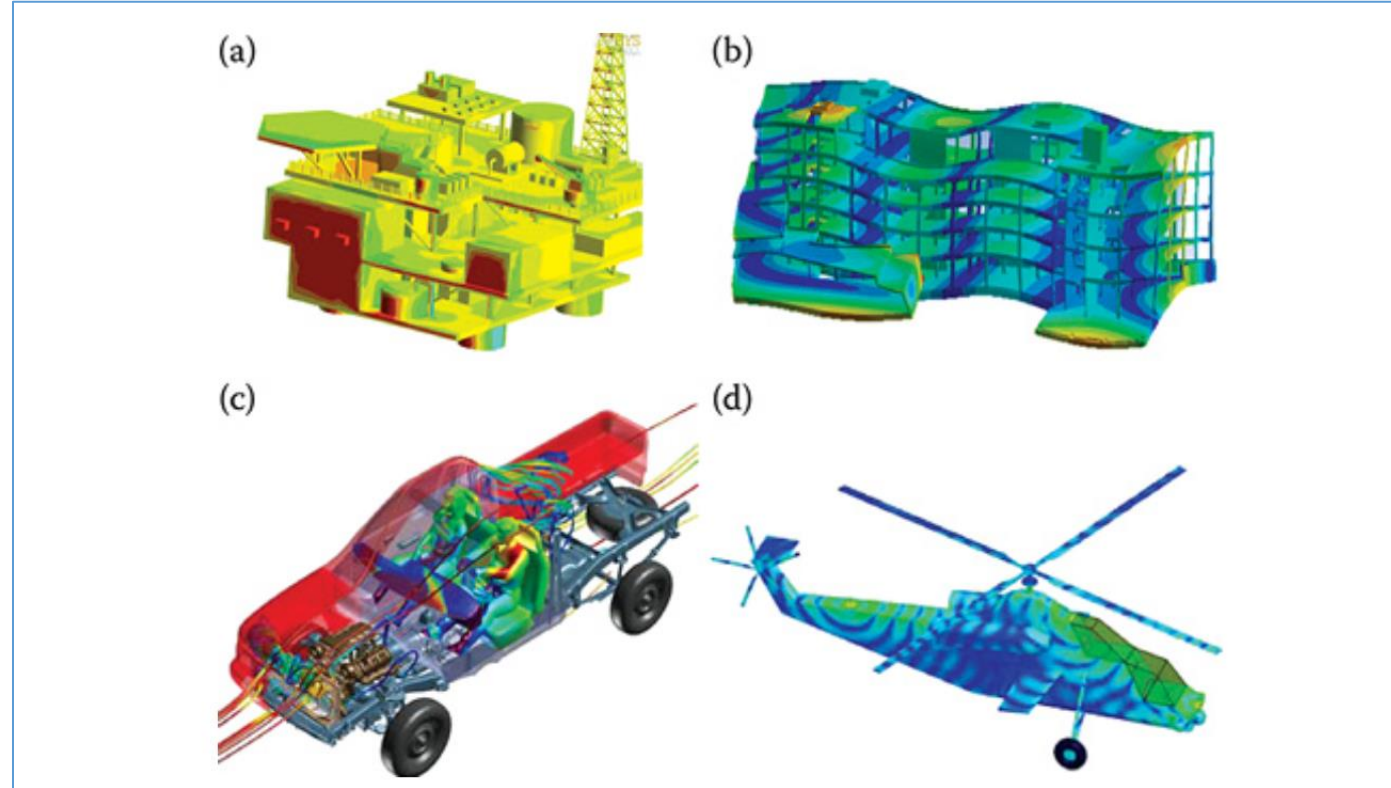
| Field of Study                  | Examples of Engineering Applications  |
|---------------------------------|---|
| Structural and solid mechanics  | Offshore structure reliability analysis, vehicle crash simulation, nuclear reactor component integrity analysis, wind turbine blade design optimization |
| Heat transfer                   | Electronics cooling modeling, casting modeling, combustion engine heat-transfer analysis  |
| Fluid flow                      | Aerodynamic analysis of race car designs, modeling of airflow patterns in buildings, seepage analysis through porous media                              |
| Electrostatics/electromagnetics | Field calculations in sensors and actuators, performance prediction of antenna designs, electromagnetic interference suppression analysis               |

### 1.1.3 FEA with ANSYS Workbench

- ❑ Many commercial programs have become available for conducting the FEA.
- ❑ Among those is ANSYS® Workbench, it is a user-friendly platform designed to seamlessly integrate ANSYS, Inc.'s suite of advanced engineering simulation technology.
- ❑ It offers bidirectional connection to major CAD systems.

## 1.1.3 FEA with ANSYS Workbench

- ❑ Workbench environment is geared toward improving productivity and ease of use among engineering teams.
- ❑ It has evolved as an indispensable tool for product development at a growing number of companies, finding applications in many diverse engineering fields (Figure 1.3).



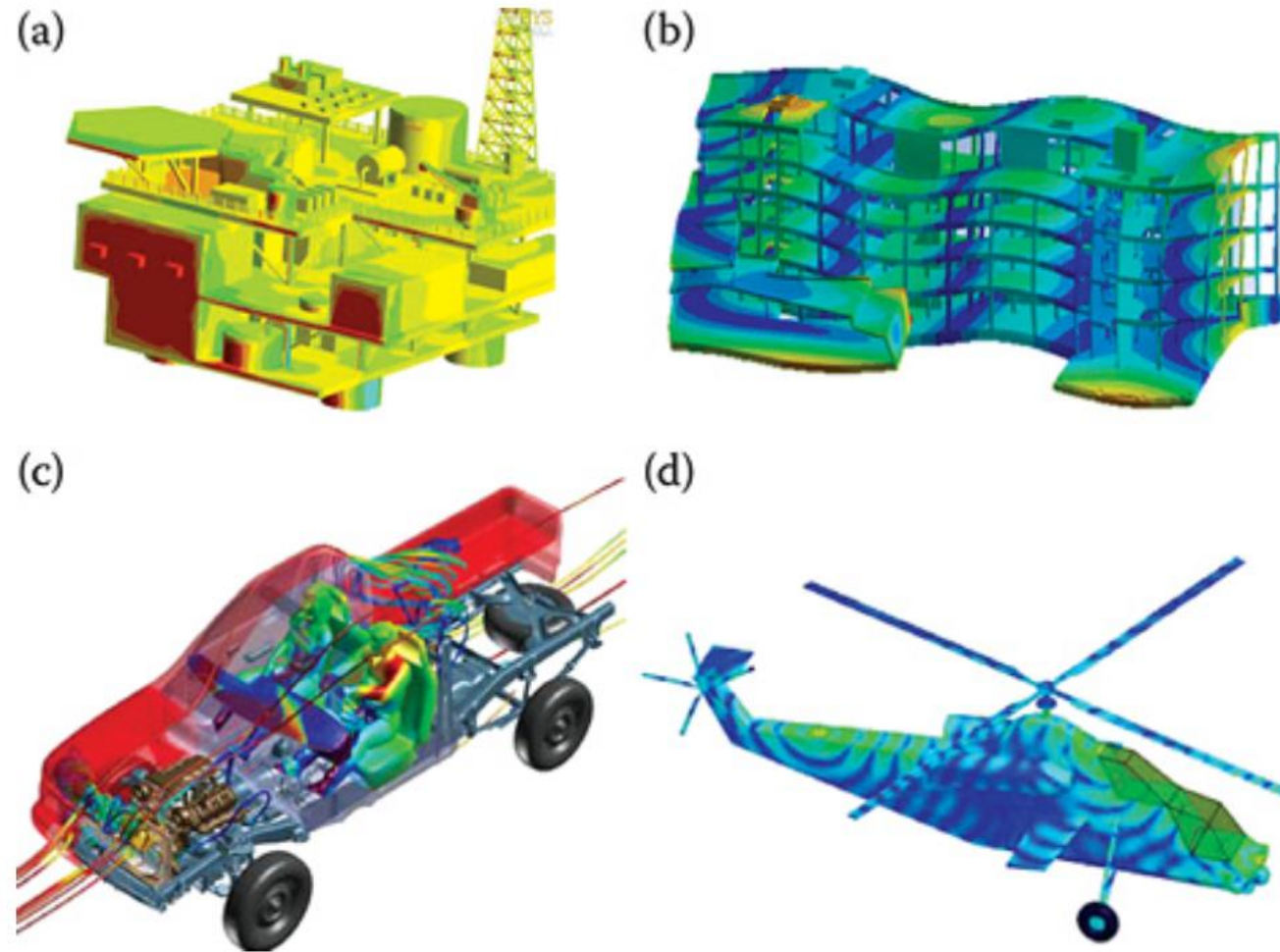


Figure 1.3 Examples of FEA using ANSYS Workbench: (a) wind load simulation of an offshore platform (Courtesy of ANSYS, Inc., <http://www.ansys.com/Industries/Energy/Oil+&+Gas>); (b) modal response of a steel frame building with concrete slab floors (<http://www.isvr.co.uk/modelling/>); (c) under hood flow and thermal management (Courtesy of ANSYS, Inc.,



## 1.1.5 A General Procedure for FEA

To conduct an FEA, the following procedure is required in general:

1. Divide the CAD/geometric model into pieces to create a “mesh” (a collection of elements with nodes, Figure 1.4).
2. Describe the behavior of the physical quantities on each element.
3. Connect (assemble) the elements at the nodes to form an approximate system of equations for the entire model.
4. Apply loads and boundary conditions (e.g., to prevent the model from moving).
5. Solve the system of equations involving unknown quantities at the nodes (e.g., the displacements).
6. Calculate the desired quantities (e.g., strains and stresses) at elements or nodes.

In commercial FEA software, this procedure is typically rearranged into the following phases:

- ❑ **Preprocessing** (build FEM models, define element properties, and apply loads and constraints)
- ❑ **FEA solver** (assemble and solve the FEM system of equations, calculate element results)
- ❑ **Postprocessing** (sort and display the results)

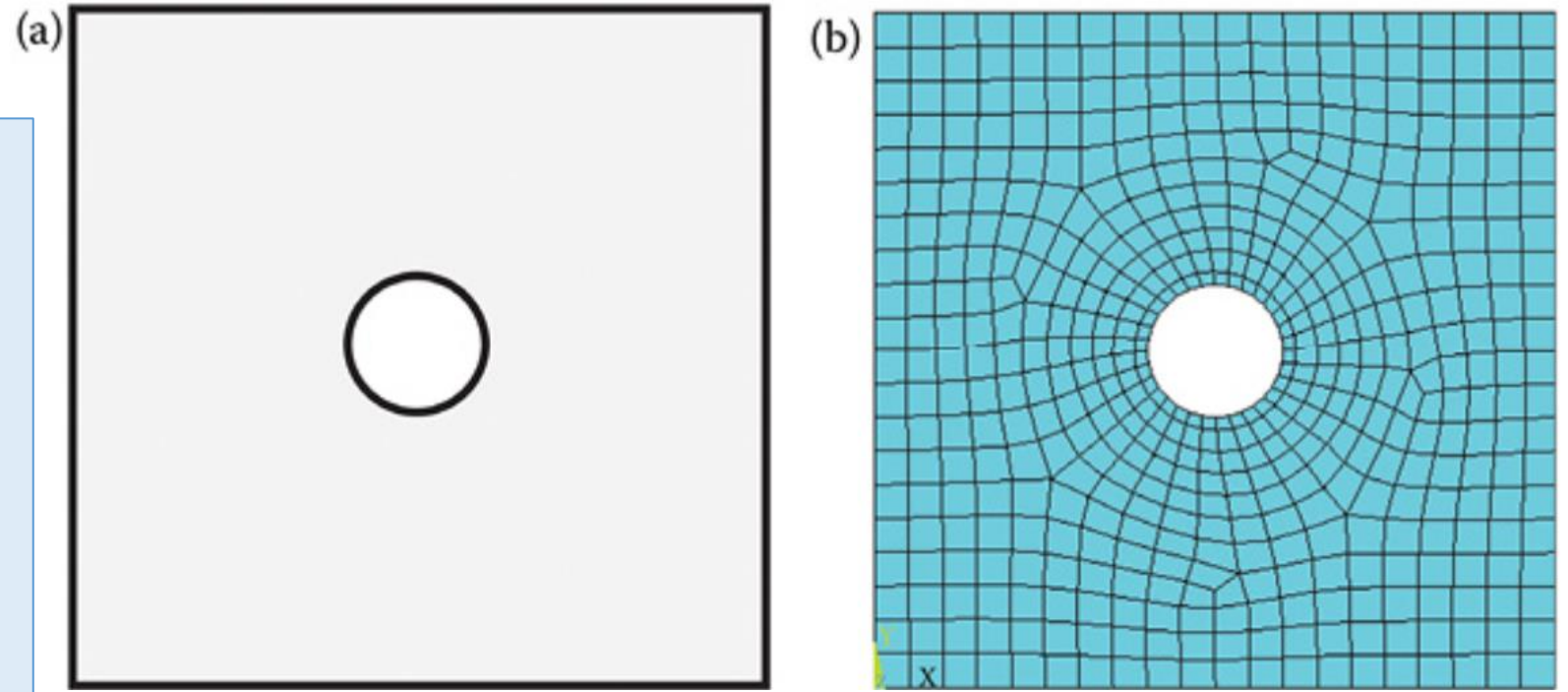


Figure 1.4(a) A plate with a hole (CAD model); and (b) A FEM discretization (mesh).



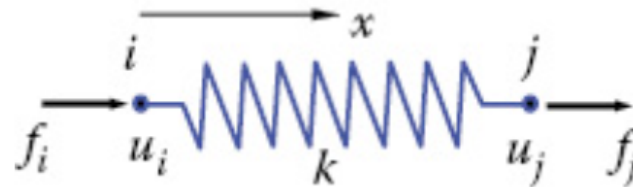
## 1.2 An Example in FEA: Spring System

A glimpse into the steps involved in an FEA is provided through a simple example in this section. We will look at a spring element and a spring system to gain insight into the basic concepts of the FEM.

### 1.2.1 One Spring Element

For the single element shown in Figure 1.5, we have:

|                                 |                     |
|---------------------------------|---------------------|
| Two nodes                       | $i, j$              |
| Nodal displacements             | $u_i, u_j$ (m, mm)  |
| Nodal forces                    | $f_i, f_j$ (Newton) |
| Spring constant (stiffness) $k$ | (N/m, N/mm)         |

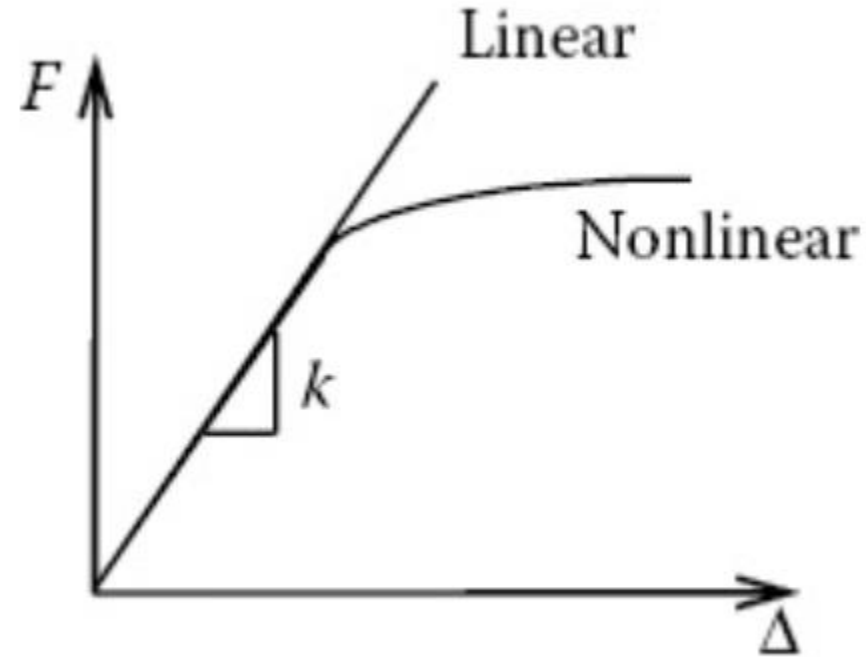


**FIGURE 1.5**

One spring element.



Relationship between spring force  $F$  and elongation  $\Delta$  is shown in Figure 1.6.



**FIGURE 1.6**

Force–displacement relation in a spring.



In the linear portion of the curve shown in [Figure 1.6](#), we have

$$F = k\Delta, \text{ with } \Delta = u_j - u_i \quad (1.1)$$

where  $k = F/\Delta (>0)$  is the stiffness of the spring (the force needed to produce a unit stretch).

Consider the equilibrium of forces for the spring. At node  $i$ , we have

$$f_i = -F = -k(u_j - u_i) = ku_i - ku_j$$



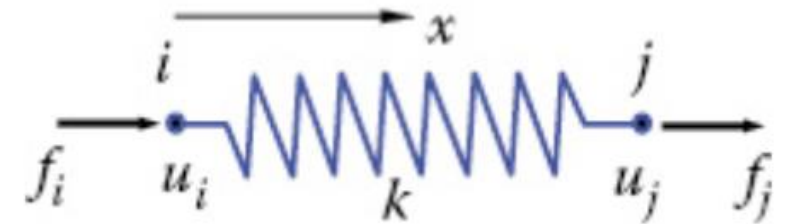
and at node  $j$

$$f_j = F = k(u_j - u_i) = -ku_i + ku_j$$



In matrix form,

$$(1.2) \quad \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i \\ f_j \end{Bmatrix}$$



or,

$$\mathbf{k}\mathbf{u} = \mathbf{f} \quad (1.3)$$

where

$\mathbf{k}$  = element stiffness matrix

$\mathbf{u}$  = element nodal displacement vector

$\mathbf{f}$  = element nodal force vector

From the derivation, we see that the first equation in [Equation 1.2](#) represents the equilibrium of forces at node  $i$ , while the second equation in [Equation 1.2](#) represents the equilibrium of forces at node  $j$ . Note also that  $\mathbf{k}$  is symmetric. Is  $\mathbf{k}$  singular or nonsingular? That is, can we solve the equation in [Equation 1.2](#)? If not, why?

### 1.2.2 A Spring System

For a system of multiple spring elements, we first write down the stiffness equation for each spring and then “assemble” them together to form the stiffness equation for the whole system. For example, for the two-spring system shown in Figure 1.7, we proceed as follows:

For element 1, we have

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \end{Bmatrix} \quad (1.4)$$

and for element 2,

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_2^2 \\ f_3^2 \end{Bmatrix} \quad (1.5)$$

where  $f_i^m$  is the (internal) force acting on *local* node  $i$  of element  $m$  ( $i = 1, 2$ ).

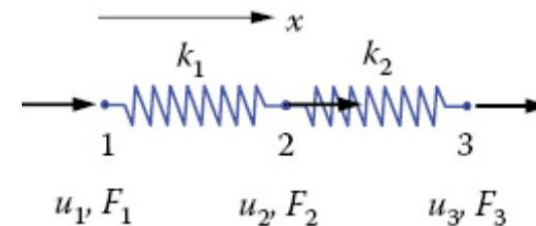


FIGURE 1.7

A system of two spring elements.

Figure 1.7

A system of two spring elements.

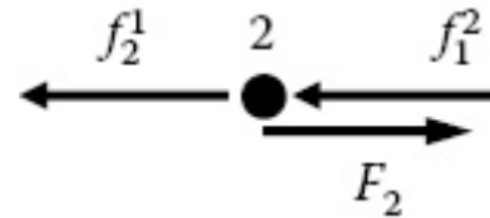
### 1.2.2.1 Assembly of Element Equations: Direct Approach

Consider the equilibrium of forces at node 1,

$$F_1 = f_1^1$$



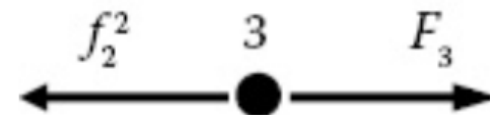
at node 2,



$$F_2 = f_2^1 + f_1^2$$

and, at node 3,

$$F_3 = f_2^2$$



Using Equations 1.4 and 1.5, we obtain

$$F_1 = k_1 u_1 - k_1 u_2$$

$$F_2 = -k_1 u_1 + (k_1 + k_2) u_2 - k_2 u_3$$

$$F_3 = -k_2 u_2 + k_2 u_3$$

In matrix form, we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

or

$$\mathbf{K}\mathbf{u} = \mathbf{F}$$

in which,  $\mathbf{K}$  is the stiffness matrix (structure matrix) for the entire spring system.

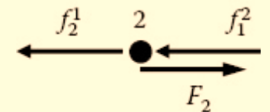
### 1.2.2.1 Assembly of Element Equations: Direct Approach

Consider the equilibrium of forces at node 1,

$$F_1 = f_1^1$$



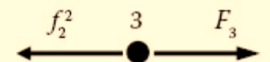
at node 2,



$$F_2 = f_2^1 + f_1^2$$

and, at node 3,

$$F_3 = f_2^2$$



(1.6)

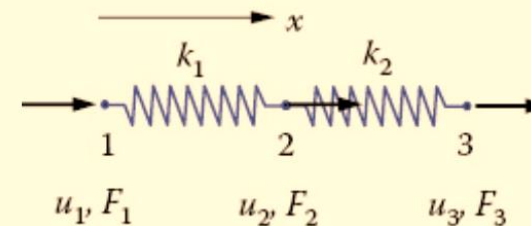


FIGURE 1.7

A system of two spring elements.

(1.7)

### 1.2.2.1.1 An Alternative Way of Assembling the Whole Stiffness Matrix

“Enlarging” the stiffness matrices for elements 1 and 2, we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 \\ 0 \end{Bmatrix}$$

and

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_1^2 \\ f_2^2 \end{Bmatrix}$$

Adding the two matrix equations (i.e., using *superposition*), we have

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} f_1^1 \\ f_2^1 + f_1^2 \\ f_2^2 \end{Bmatrix}$$

This is the same equation we derived by using the concept of equilibrium of forces.



### 1.2.2.2 Assembly of Element Equations: Energy Approach

We can also obtain the result using an energy method, for example, the principle of minimum potential energy. In fact, the energy approach is more general and considered the foundation of the FEM. To proceed, we consider the strain energy  $U$  stored in the spring system shown in [Figure 1.5](#).

$$U = \frac{1}{2} k_1 \Delta_1^2 + \frac{1}{2} k_2 \Delta_2^2 = \frac{1}{2} \Delta_1^T k_1 \Delta_1 + \frac{1}{2} \Delta_2^T k_2 \Delta_2$$

However,

$$\Delta_1 = u_2 - u_1 = [-1 \quad 1] \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}, \quad \Delta_2 = u_3 - u_2 = [-1 \quad 1] \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$

We have

$$\begin{aligned} U &= \frac{1}{2} [u_1 \quad u_2] \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} + \frac{1}{2} [u_2 \quad u_3] \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = (\text{enlarging } \dots) \\ &= \frac{1}{2} [u_1 \quad u_2 \quad u_3] \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \end{aligned} \quad (1.8)$$



The potential of the external forces is

$$\Omega = -F_1 u_1 - F_2 u_2 - F_3 u_3 = -[u_1 \quad u_2 \quad u_3] \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \quad (1.9)$$

Thus, the total potential energy of the system is

$$\Pi = U + \Omega = \frac{1}{2} [u_1 \quad u_2 \quad u_3] \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} - [u_1 \quad u_2 \quad u_3] \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \quad (1.10)$$

which is a function of the three nodal displacements  $(u_1, u_2, u_3)$ . According to the principle of minimum potential energy, for a system to be in equilibrium, the total potential energy must be minimum, that is,  $d\Pi = 0$ , or equivalently,

$$\frac{\partial \Pi}{\partial u_1} = 0, \quad \frac{\partial \Pi}{\partial u_2} = 0, \quad \frac{\partial \Pi}{\partial u_3} = 0, \quad (1.11)$$

which yield the same three equations as in [Equation 1.6](#).



### 1.2.3 Boundary and Load Conditions

Assuming that node 1 is fixed, and same force  $P$  is applied at node 2 and node 3, that is

$$u_1 = 0 \text{ and } F_2 = F_3 = P$$

we have from Equation 1.6

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ P \end{Bmatrix}$$

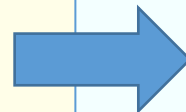
which reduces to

$$\begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P \\ P \end{Bmatrix}$$

and

Unknowns are

$$\mathbf{u} = \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$



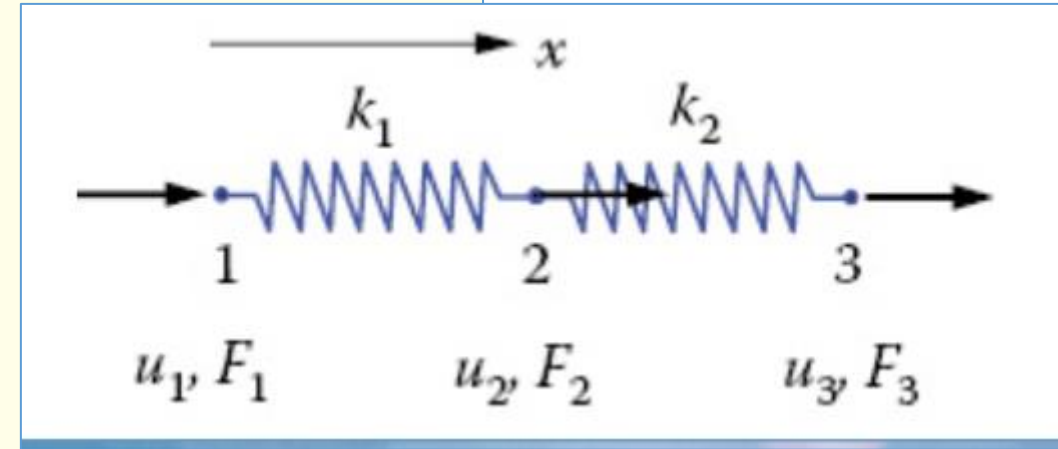
and the reaction force  $F_1$  (if desired).

Solving the equations, we obtain the displacements

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 2P/k_1 \\ 2P/k_1 + P/k_2 \end{Bmatrix}$$

and the reaction force

$$F_1 = -2P$$



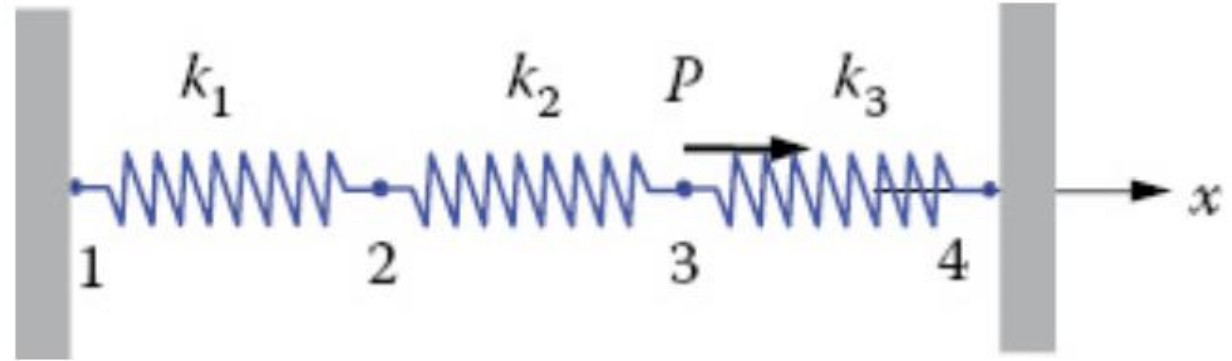
## 1.2.4 Solution Verification

It is **very important** in FEA to verify the results you obtained through either hand calculations or analytical solutions in the literature. The following is a list of items to check based on common sense or intuition, or analytical solutions if they are available.

- Deformed shape of the structure
  - Equilibrium of the external forces (Reaction forces should balance with the applied loads.)
  - Order of magnitudes of the obtained values
- Notes about the Spring Elements:
- Spring elements are only suitable for stiffness analysis.
  - They are not suitable for stress analysis of the spring itself.
  - There are spring elements with stiffness in the lateral direction, spring elements for torsion, and so on.

## 1.2.5 Example Problems

### EXAMPLE 1.1



*Given:* For the spring system shown above,

$$k_1 = 100 \text{ N/mm}, \quad k_2 = 200 \text{ N/mm}, \quad k_3 = 100 \text{ N/mm}$$

$$P = 500 \text{ N}, \quad u_1 = u_4 = 0$$

*Find:*

- The global stiffness matrix
- Displacements of nodes 2 and 3
- The reaction forces at nodes 1 and 4
- The force in the spring 2



## Solution

a. The element stiffness matrices are (make sure to put proper unit after each number)

$$\mathbf{k}_1 = \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \text{ (N/mm)}$$

$$\mathbf{k}_2 = \begin{bmatrix} 200 & -200 \\ -200 & 200 \end{bmatrix} \text{ (N/mm)}$$

$$\mathbf{k}_3 = \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \text{ (N/mm)}$$

Applying the superposition concept, we obtain the global stiffness matrix for the spring system

$$\mathbf{K} = \begin{matrix} & \begin{matrix} u_1 & u_2 & u_3 & u_4 \end{matrix} \\ \begin{matrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{matrix} & \begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 100 + 200 & -200 & 0 \\ 0 & -200 & 200 + 100 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix} \end{matrix}$$

or

$$\mathbf{K} = \begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 300 & -200 & 0 \\ 0 & -200 & 300 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix}$$

which is *symmetric* and *banded*.

Equilibrium (FE) equation for the whole system is

$$\begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 300 & -200 & 0 \\ 0 & -200 & 300 & -100 \\ 0 & 0 & -100 & 100 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix}$$

- b. Applying the BCs  $u_1 = u_4 = 0$ ,  $F_2 = 0$ , and  $F_3 = P$ , and “deleting” the first and fourth rows and columns, we have

$$\begin{bmatrix} 300 & -200 \\ -200 & 300 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ P \end{Bmatrix}$$

Solving this equation, we obtain

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} P/250 \\ 3P/500 \end{Bmatrix} = \begin{Bmatrix} 2 \\ 3 \end{Bmatrix} (\text{mm})$$

- c. From the first and fourth equations in the system of FE equations, we obtain the reaction forces

$$F_1 = -100u_2 = -200 \text{ (N)}$$

$$F_4 = -100u_3 = -300 \text{ (N)}$$



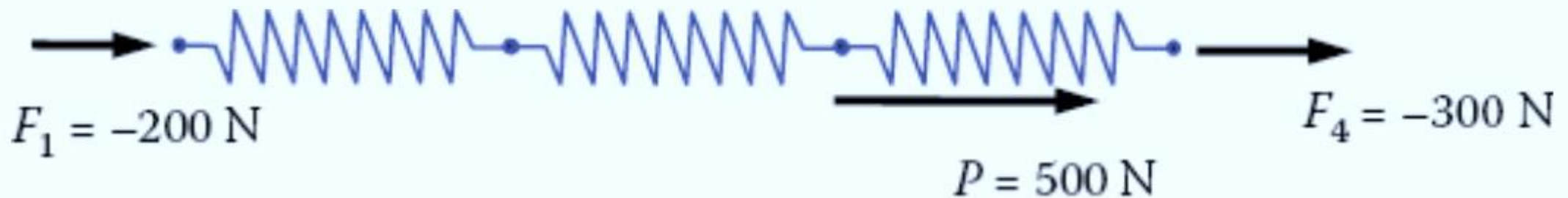


Here  $i = 2, j = 3$  for element 2. Thus we can calculate the spring force as

$$F = f_j = -f_i = [-200 \quad 200] \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = [-200 \quad 200] \begin{Bmatrix} 2 \\ 3 \end{Bmatrix} = 200 \text{ (N)}$$

*Check the results:*

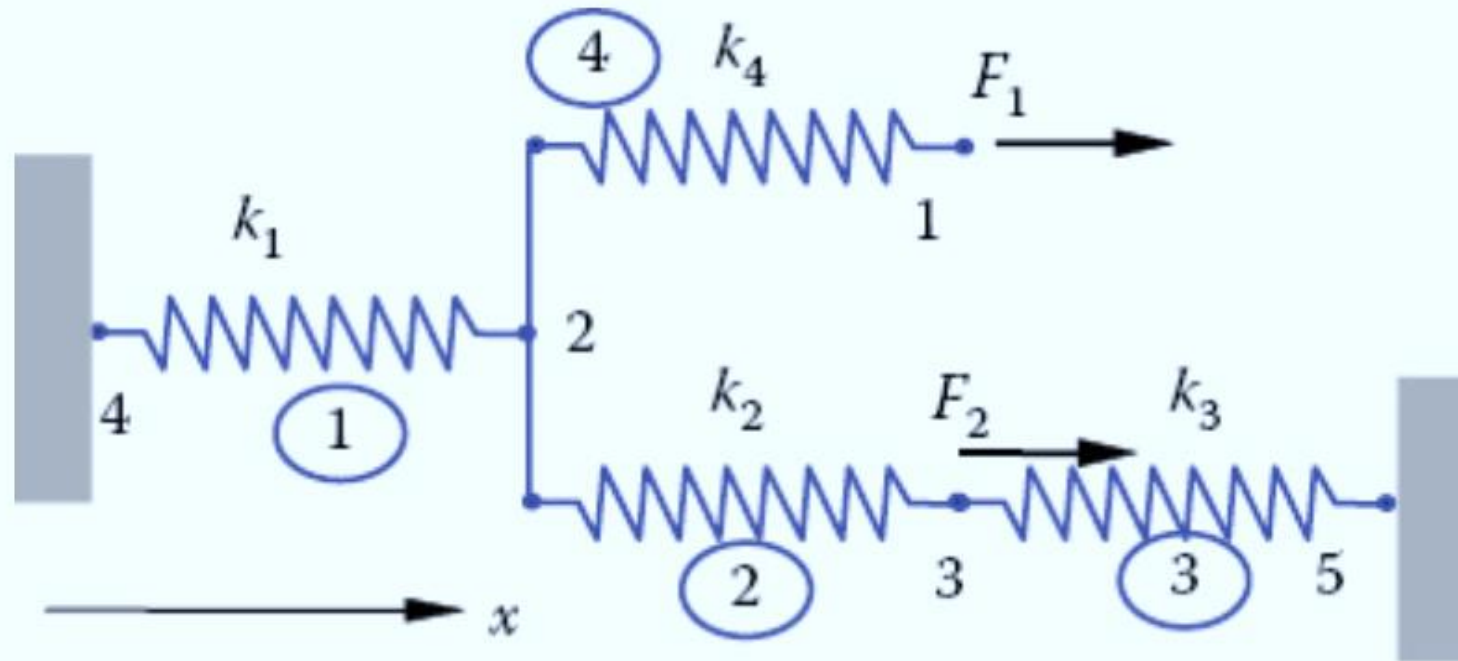
Draw the free-body diagram (FBD) of the system and consider the equilibrium of the forces.



Equilibrium of the forces is satisfied!



## EXAMPLE 1.2



### Problem

For the spring system with arbitrarily numbered nodes and elements, as shown above, find the global stiffness matrix.

## Solution

First, we construct the following *element connectivity table*:

| Element | Node $i$ (1) | Node $j$ (2) |
|---------|--------------|--------------|
| 1       | 4            | 2            |
| 2       | 2            | 3            |
| 3       | 3            | 5            |
| 4       | 2            | 1            |

This table specifies the *global* node numbers corresponding to the *local* node numbers for each element.

This table specifies the *global* node numbers corresponding to the *local* node numbers for each element.

Then we write the element stiffness matrix for each element

$$\mathbf{k}_1 = \begin{matrix} & \begin{matrix} u_4 & u_2 \end{matrix} \\ \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix}, \end{matrix}$$

$$\mathbf{k}_2 = \begin{matrix} & \begin{matrix} u_2 & u_3 \end{matrix} \\ \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}, \end{matrix}$$

$$\mathbf{k}_3 = \begin{matrix} & \begin{matrix} u_3 & u_5 \end{matrix} \\ \begin{bmatrix} k_3 & -k_3 \\ -k_3 & k_3 \end{bmatrix}, \end{matrix}$$

$$\mathbf{k}_4 = \begin{matrix} & \begin{matrix} u_2 & u_1 \end{matrix} \\ \begin{bmatrix} k_4 & -k_4 \\ -k_4 & k_4 \end{bmatrix} \end{matrix}$$

Finally, applying the superposition method, we obtain the global stiffness matrix as follows:

Finally, applying the superposition method, we obtain the global stiffness matrix as follows:

$$\mathbf{K} = \begin{matrix} & \begin{matrix} u_1 & u_2 & u_3 & u_4 & u_5 \end{matrix} \\ \begin{matrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{matrix} & \begin{bmatrix} k_4 & -k_4 & 0 & 0 & 0 \\ -k_4 & k_1 + k_2 + k_4 & -k_2 & -k_1 & 0 \\ 0 & -k_2 & k_2 + k_3 & 0 & -k_3 \\ 0 & -k_1 & 0 & k_1 & 0 \\ 0 & 0 & -k_3 & 0 & k_3 \end{bmatrix} \end{matrix}$$

The matrix is *symmetric*, *banded*, but *singular*, as it should be.

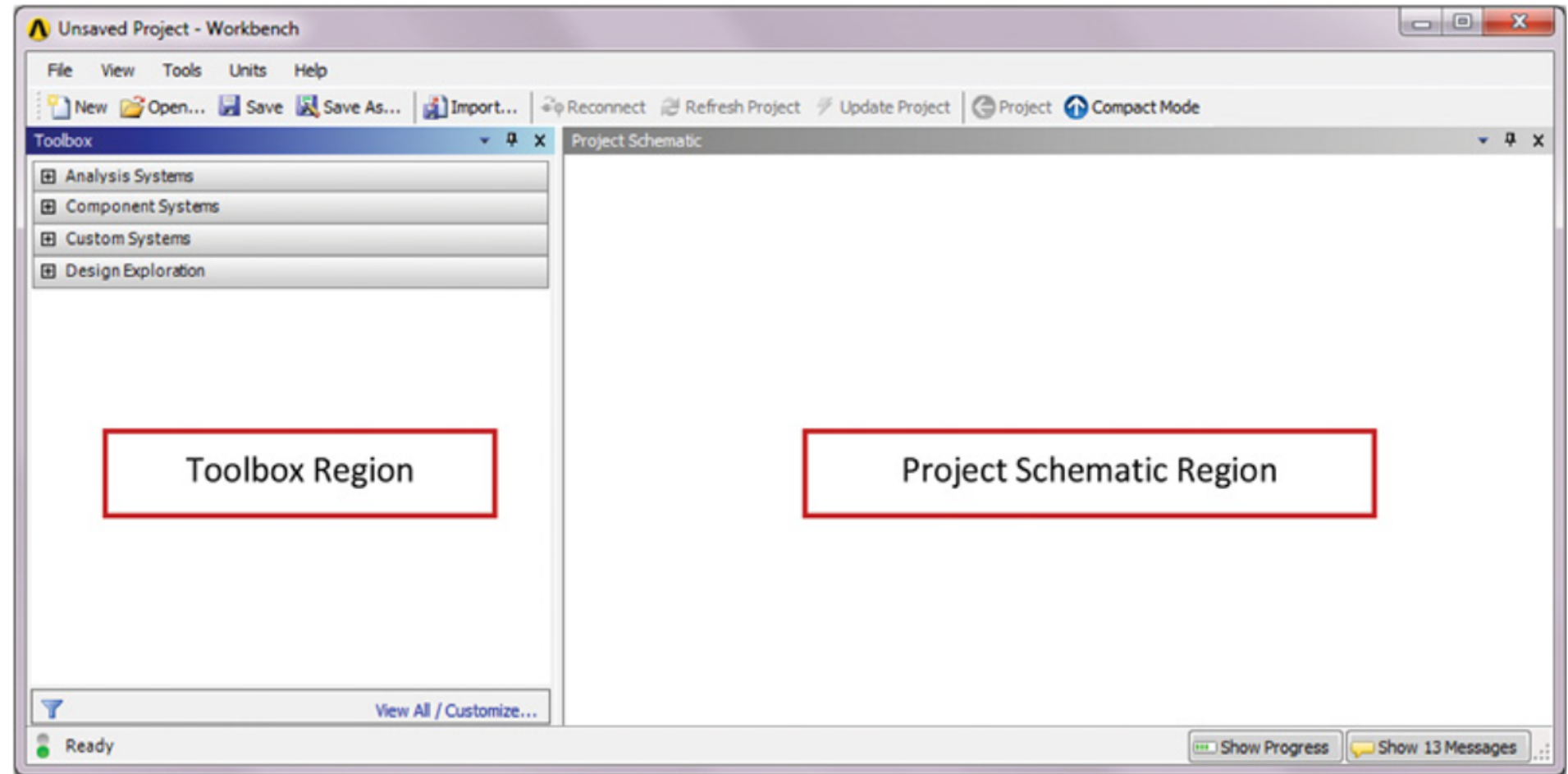
After introducing the basic concepts, the section below introduces you to one of the general-purpose finite element software tools—ANSYS Workbench.

## 1.3 Overview of ANSYS Workbench

- ANSYS Workbench is a simulation platform that enables users to model and solve a wide range of engineering problems using the FEA.
- It provides access to the ANSYS family of design and analysis modules in an integrated simulation environment.
- This section gives a brief overview of the different elements in the ANSYS Workbench simulation environment or the graphical-user interface (GUI).
- Readers are referred to ANSYS Workbench user's guide [2] for more detailed information.

### 1.3.1 The User Interface

The *Workbench* interface is composed primarily of a *Toolbox* region and a *Project Schematic* region (Figure 1.8). The main use of the two regions is described next.



**FIGURE 1.8**

*ANSYS Workbench* user interface.



## 1.3.2 The Toolbox

The Toolbox contains the following four groups of systems:

- ❑ **Analysis Systems:** Predefined analysis templates to be used to build your project, including static structural, steady-state thermal, transient thermal, fluid flow, modal, shape optimization, linear buckling, and many others.
- ❑ **Component Systems:** Component applications that can be used to build or expand an analysis system, including geometry import, engineering data, mesh, postprocessing, and others.
- ❑ **Custom Systems:** Coupled-field analysis systems such as fluid solid interaction, prestress modal, thermal-stress, and others.
- ❑ **Design Exploration:** Parametric optimization studies such as response surface optimization, parameters correlation, six sigma analysis, and others.



### 1.3.3 The Project Schematic

- ❑ A project schematic, that is, a graphical representation of the workflow, can be built by dragging predefined analysis templates or other components from the Toolbox and dropping them into the Project Schematic window.
- ❑ “Drag” here means to move the mouse while holding down the left mouse button, and “drop” means to release the mouse button.
- ❑ To build a project for static structural analysis, for instance, drag the Static Structural template from the Toolbox and drop it into the rectangular box that appears in the Project Schematic window.
- ❑ A standalone analysis system that contains the components needed for static structural analysis is added to the project schematic as shown in Figure 1.9a. The system consists of seven individual components called cells.

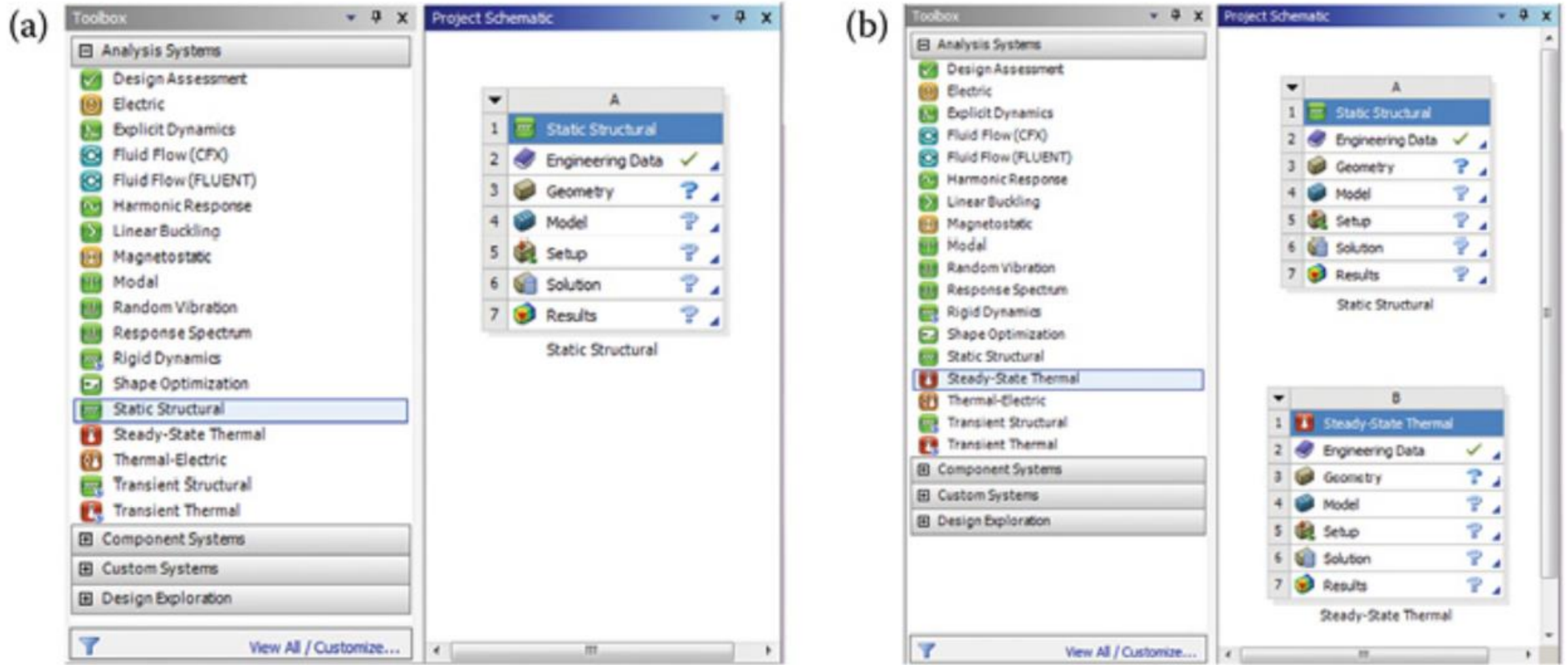


Figure 1.9 Defining standalone analysis systems in the project schematic:  
(a) a standalone system; (b) two independent standalone systems

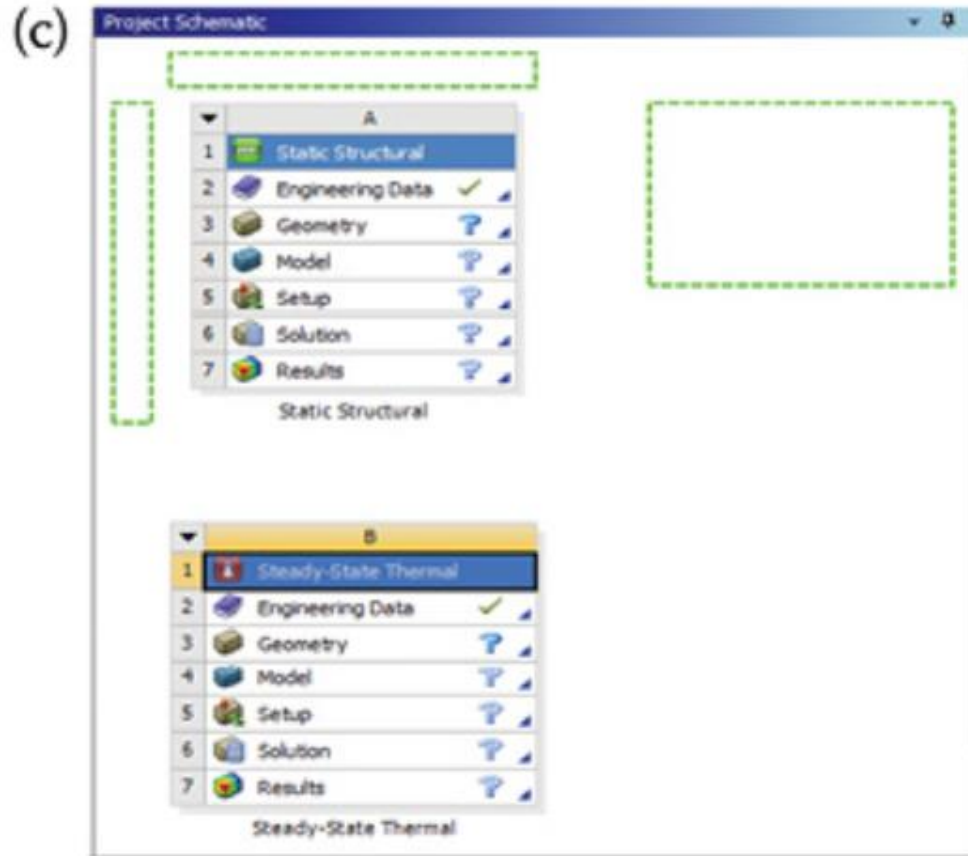


Figure 1.9 Defining standalone analysis systems in the project schematic;  
(c) moving a system in a top-bottom configuration; and (d) moving a system in a side-by-side configuration.

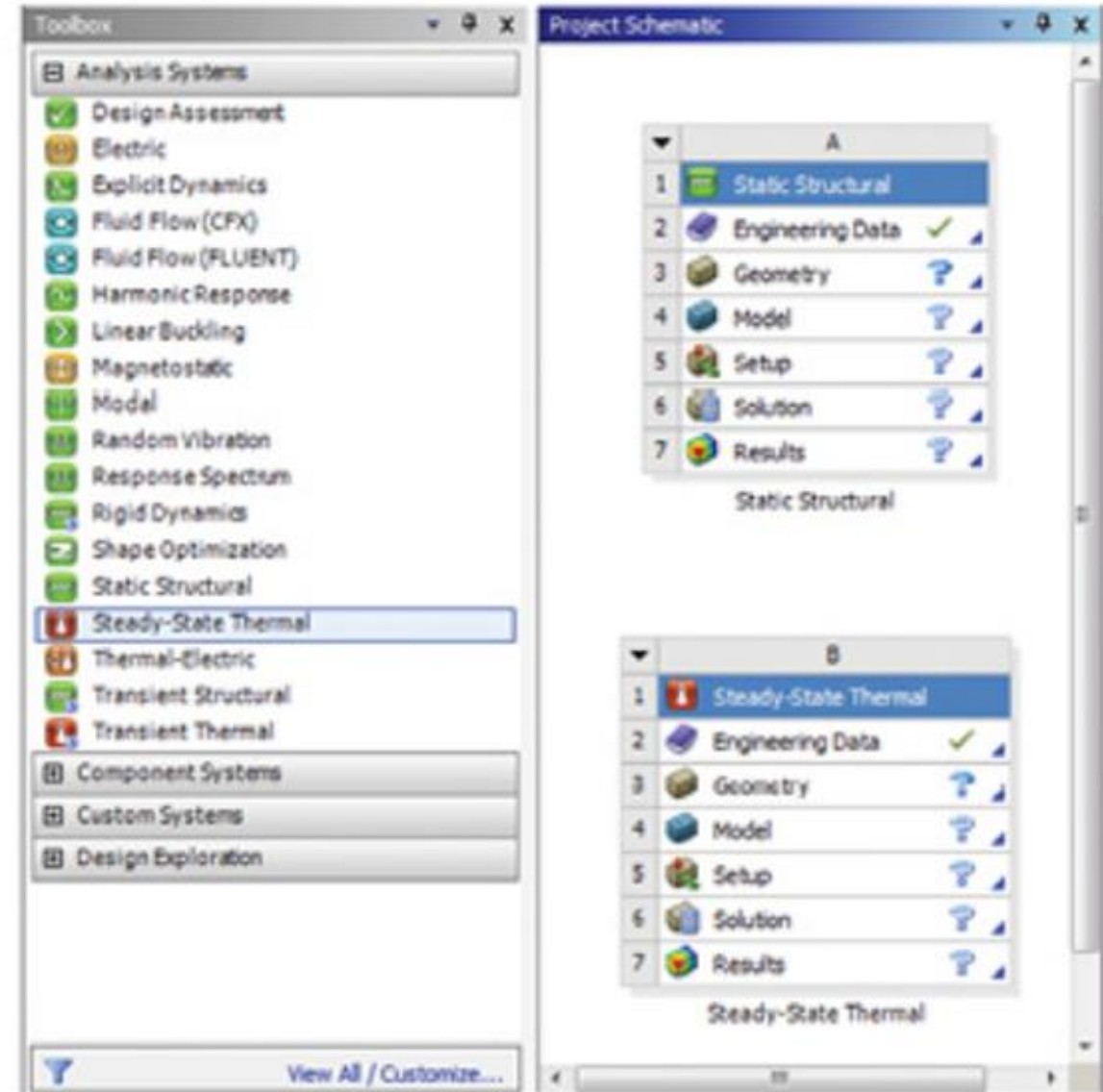
Alternatively, a standalone analysis can be created by double-clicking.

For example, double-click the **Steady-State Thermal** template from the **Toolbox**, and an independent Steady-State Thermal system will be placed in the default location below the existing Static Structural system, as shown in Figure 1.9b.



To delete a system, click on the down arrow button at the upper left corner of the system from the Project Schematic window, and then choose Delete from the drop-down context menu.

(b)





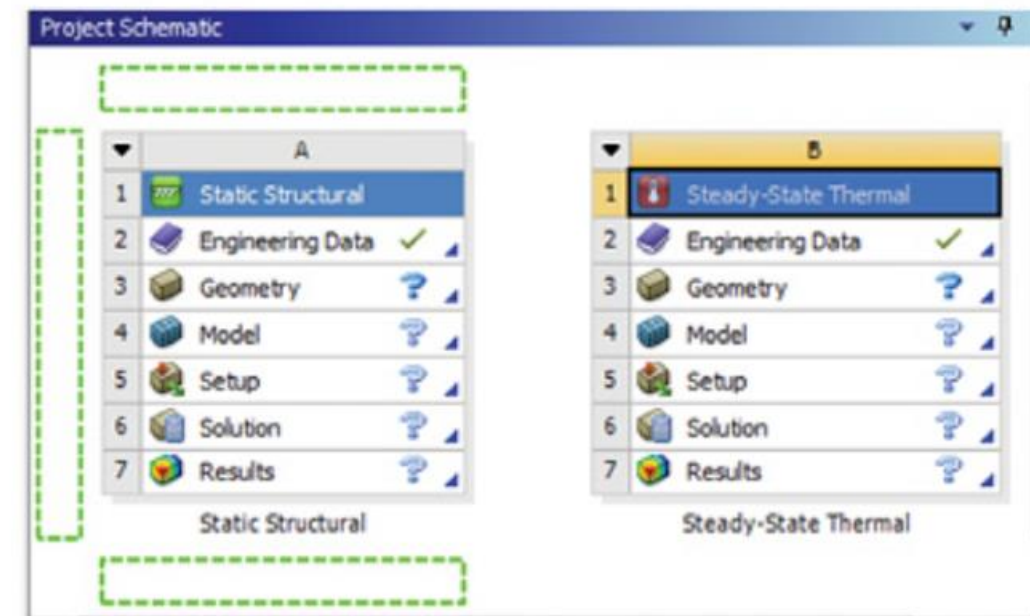
A system can be moved around another system in the project schematic. To move a system, click on the header cell (i.e., the cell titled Steady-State Thermal for the thermal system) and drag it to a new place.

Once you drag the header cell, dashed rectangles appear for the possible new locations to drop the system. This is illustrated in Figures 1.9 c and d for two systems with initial top-bottom and side-by-side configurations, respectively.

To delete a system, click on the down arrow button at the upper left corner of the system from the Project Schematic window, and then choose Delete from the drop-down context menu.



(d)



In some cases, a project may contain two or more analysis systems that share data.

For example, a downstream modal analysis may use the same material, geometry, and model data from the preceding structural analysis.

To build such a project, create a standalone system for Static Structural analysis. Then, drag the Modal analysis template from the Toolbox and drop it onto the Model cell of the Static Structural system.

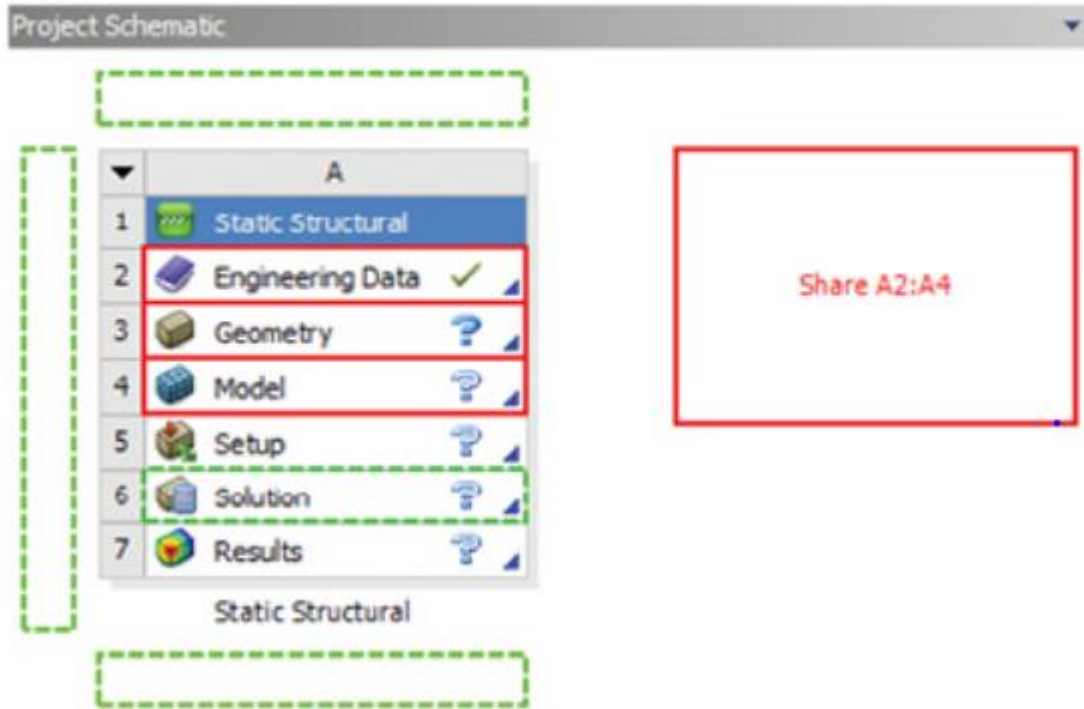
Immediately before the subsequent system is dropped, **bounding boxes will appear** on the Engineering Data, Geometry, and Model cells of the first system, as shown in Figure 1.10a



Figure 1.10 Defining linked analysis systems in the project schematic: (a) (a) dropping the second (subsequent) system onto the Model cell of the first system to share data at the model and above levels



(a)



(b)

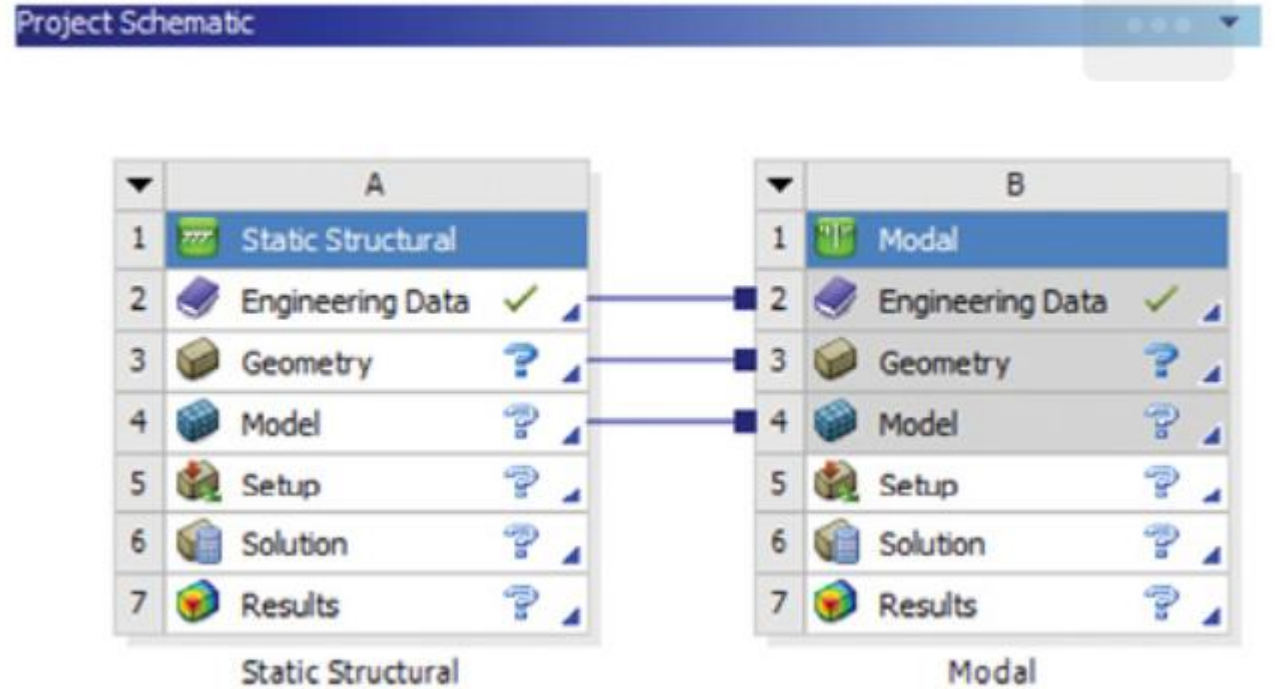


Figure 1.10 Defining linked analysis systems in the project schematic: (a) dropping the second (subsequent) system onto the Model cell of the first system to share data at the model and above levels; (b) two systems that are linked.

In some cases, a project may contain two or more analysis systems that share data.

For example, a downstream modal analysis may use the same material, geometry, and model data from the preceding structural analysis.

To build such a project, create a standalone system for Static Structural analysis. Then, drag the Modal analysis template from the Toolbox and drop it onto the Model cell of the Static Structural system.

Immediately before the subsequent system is dropped, bounding boxes will appear on the Engineering Data, Geometry, and Model cells of the first system, as shown in Figure 1.10a.

After the system is released, a project including two linked systems is created, as shown in Figure 1.10b, where the linked cells indicate data sharing at the Model and above levels.

## 1.3.4 Working with Cells

Cells are components that make up an analysis system. You may launch an application by double-clicking a cell. To initiate an action other than the default action, right-click on a cell to view its context menu options. The following list comprises the types of cells available in ANSYS Workbench and their intended functions:

**Engineering Data:** Define or edit material models to be used in an analysis.

**Geometry:** Create, import, or edit the geometry model used for analysis.

**Model/Mesh:** Assign material, define coordinate system, and generate mesh for the model.

**Setup:** Apply loads, boundary conditions, and configure the analysis settings.

**Solution:** Access the model solution or share solution data with other downstream systems.









**Results:** Indicate the results availability and status (also referred to as postprocessing).

As the data flows through a system, a cell's state can quickly change. ANSYS Workbench provides a state indicator icon placed on the right side of the cell. Table 1.2 describes the indicator icons and the various cell states available in ANSYS Workbench. For more information, please refer to ANSYS Workbench user's guide [2].

Table 1.2

## Indicator Icons and Descriptions of the Various Cell States

Indicator Icons and Descriptions of the Various Cell States

| Cell State            | Indicator   | Description   |
|-----------------------|---|---|
| Unfulfilled           |    | Need upstream data to proceed   |
| Refresh required      |    | A refresh action is needed as a result of changes made on upstream data   |
| Attention required    |    | User interaction with the cell is needed to proceed   |
| Update required       |    | An update action is needed as a result of changes made on upstream data   |
| Up to date            |    | Data are up to date and no attention is required  |
| Input changes pending |   | An update or refresh action is needed to recalculate based on changes made to upstream cells                      |
| Interrupted           |  | Solution has been interrupted. A resume or update action will make the solver continue from the interrupted point |
| Pending               |  | Solution is in progress   |



## 1.3.5 The Menu Bar

The menu bar is the horizontal bar anchored at the top of the Workbench user interface. It provides access to the following functions:

**File Menu:** Create a new project, open an existing project, save the current project, and so on.

**View Menu:** Control the window/workspace layout, customize the toolbox, and so on.

**Tools Menu:** Update the project and set the license preferences and other user options.

**Units Menu:** Select the unit system and specify unit display options.

**Help Menu:** Get help for ANSYS Workbench.



## 1.4 Summary

- ❑ In this chapter, the basic concepts in the FEM are introduced.
- ❑ The spring system is used as an example to show how to establish the element stiffness matrices, to assemble the finite element equations for a system from element stiffness matrices, and to solve the FE equations. Verifying the FE results is emphasized.
- ❑ ANSYS Workbench environment is briefly introduced.
- ❑ The concepts and procedures introduced in this chapter are very simple and yet very important for studying the finite element analyses of other problems.





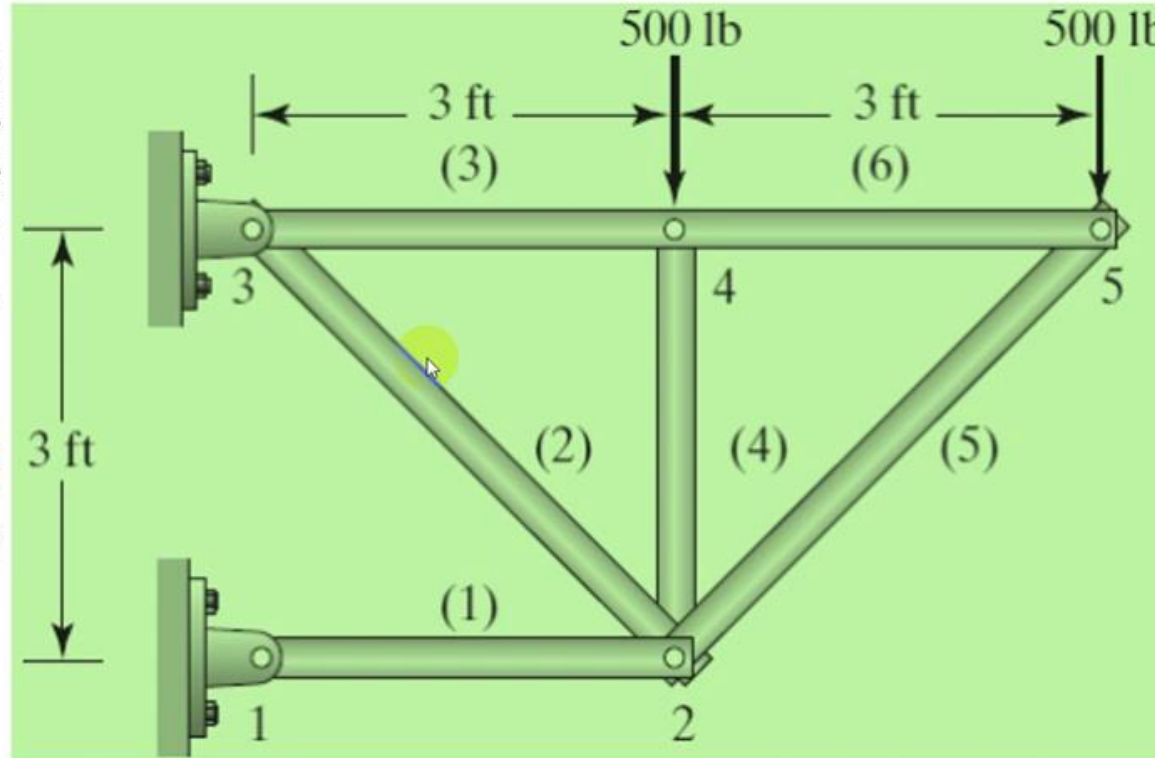
L

## 2-D Truss Analysis in ANSYS Workbench

### Balcony Truss

A balcony truss shown is in the figure, all members are made from Douglas-fir wood with a modulus of elasticity ( $E$ ) of  $1.90 \times 10^6$  lb/in<sup>2</sup>, Poisson's ratio ( $\nu$ ) of 0.3 and a cross-sectional ( $A$ ) area of 8 in<sup>2</sup> (assume rectangular cross-section, breadth as 2 in and depth as 4 in).

Determine deflection of each joint, axial force and stress for each member, and reaction at each support under loading shown in the Figure.



#### Example 3.2

Chapter 3: Trusses

*Finite Element Analysis Theory and Application with ANSYS*

Saeed Moaveni, 4<sup>th</sup> Edition

## Practice No. 1

**Part1 How to setup spring mass system in workbench Mechanical ?**

<https://www.youtube.com/watch?v=q0RG6SrBbkl>

## Practice No. 2

 **Truss Analysis Ansys Workbench**

<https://www.youtube.com/watch?v=sKZAipnrV9E>

## Practice No. 3

**2-D Balcony Truss Analysis in ANSYS Workbench**

<https://www.youtube.com/watch?v=frFSm82FTzk><https://www.youtube.com/watch?v=frFSm82FTzk>

